Introduction to CNC & CAM ENGI 7928

Getting started...

The CNC Lab

- Introduction to new software: Mastercam (2D milling)
- You will draw a model in SolidWorks - sketches must be fully defined
- You will use a tutorial handout to "machine" a part in Mastercam
- Tutorial: Mill_1_handout available in the 7928 folder on the S:\engr\courses\7928 folder
- When working in Mastercam- you must be diligent with the values you input, a typo could equal a crash.
- Mastercam Labs: Thurs., July 5, Fri. July 6 with TA available
- CNC Demonstration- in groups of 6 between July 11&12, 2-4pm
- NOTE :Mastercam is limited to 12 licenses only, therefore you will need to find time when a station is available for you to complete the lab

Memorial Engineering Student Machine Shop
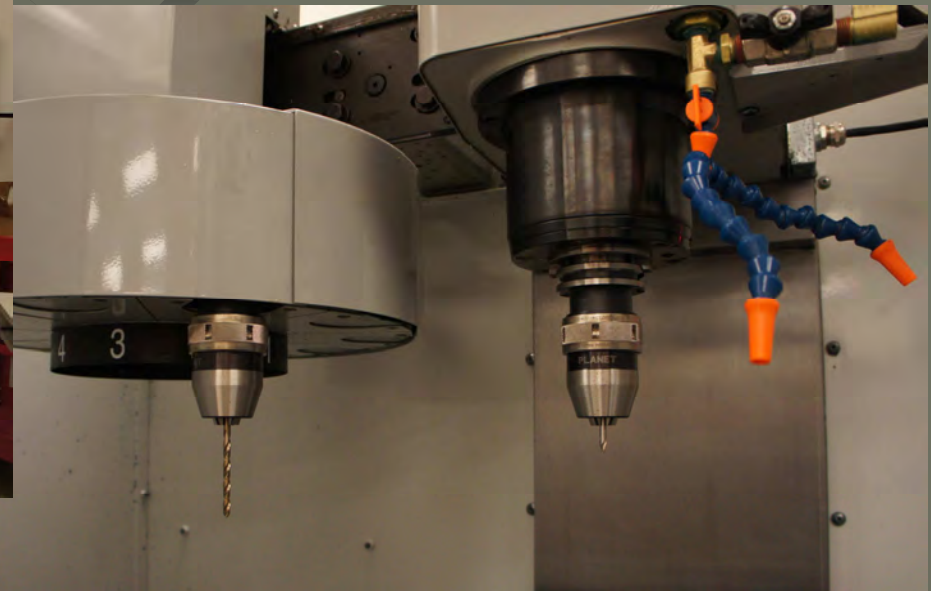
- Haas Super Mini CNC Mill, 15 hp
- 10,000 rpm, 833 ipm feed cutting rate
- 10 Pocket umbrella tool changer
- Travel: X=16", Y=12", Z=10"

Memorial Engineering
Student Machine Shop

- Mazak Quick Turn Nexus 100-II CNC Lathe, 15hp
- 6,000 rpm, 1180 ipm cutting feed rate
- 12 position tool changer
- Travel: X= 11", Z=12"

Many types of CNC machines & many outcomes!

- Lathes, Mills
- 2 to 5 Axis
- Lathe bar feeders
- Pallet Changer Centres

Photos compliments of Vincennes University www.vuhtec.org

Simple to complex

What is CNC?

- Used to automate machine tools to perform simple to complex machining operations

- Many controller languages are used i.e. conversational & G code

- Manually: code is used to build a program by manually entering commands using the key pad

- Software: for more complex or feature intense components we use CAM software to generate code for lengthy machining processes and /or complex surfaces

Writing Code

- G-Code

- Conversational

06-2012   C. Koenig
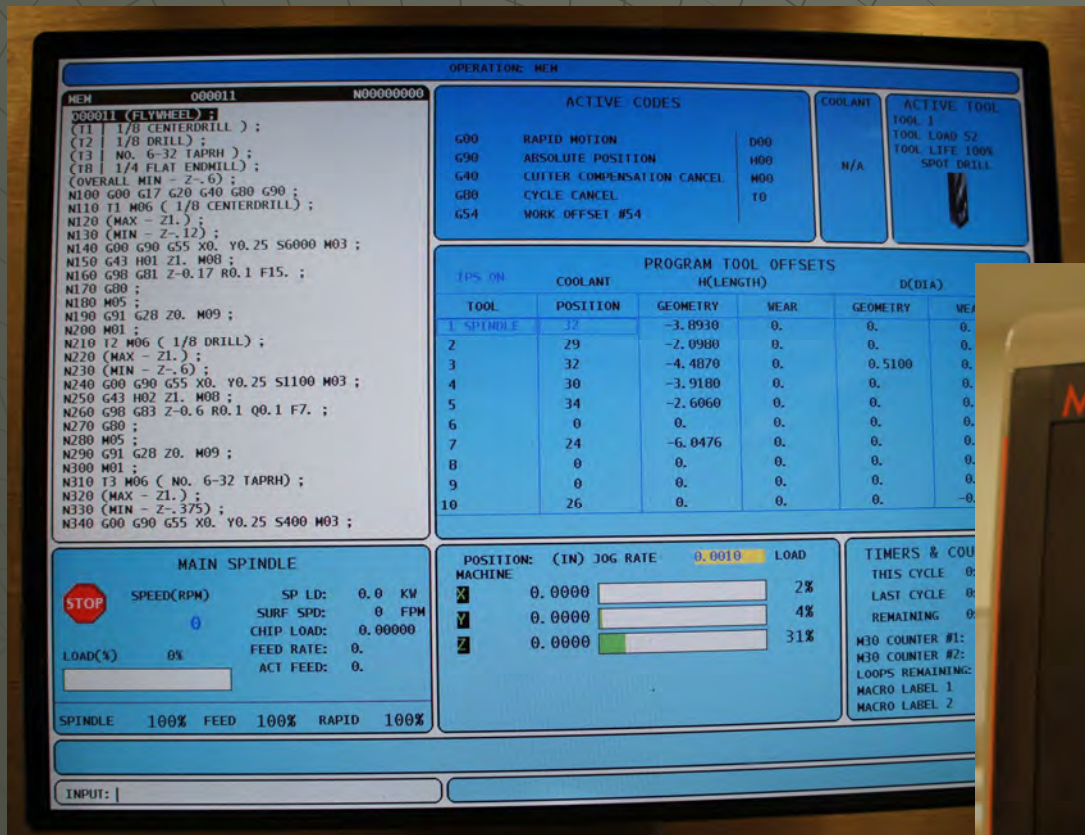
What is CAM?

- Computer Aided Manufacturing

- Used to assist manufacturers to generate CNC programs

- Not a simple "click of a button"

- CAM can be parameter heavy: tools must be selected, speeds, feeds, depth of cuts, efficient tool paths selected & verified, clearances checked, reduce program times etc.

- CAM software allows us to machine components VIRTUALLY!

7928- Using Mastercam

- Introduction to Mastercam
- Importing SolidWorks models
- Selecting stock size
- Setting up machining parameters
- Selecting machining operations, tool types, feeds, speeds
- Determine axial & radial depth cuts
- Decide on rough & finish passes
- Verify toolpaths & cycle times
- Generate tool list, set up sheet & post operations

Design a solid model in SolidWorks:
TIP: *keep it simple*

- Note: the front plane in SolidWorks will become the top plane in Mastercam.

- Use the recommended approach to drawing the solid model. In Mastercam we will make some minor modifications to reposition the origin.

- Save the SolidWorks model as an .x_t parasolid file

- **Fully define all sketches in SW - OR potential m/c CRASH!**
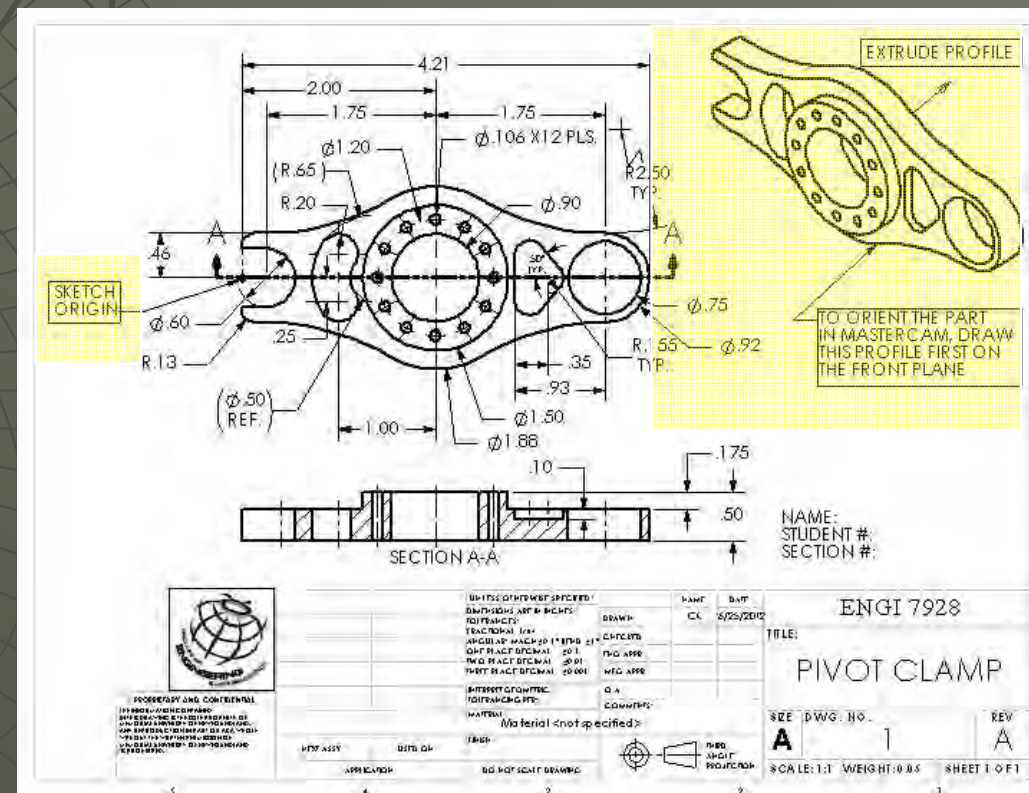
# Mastercam is limited to 12 licenses in this lab
- It is available also available in your home room
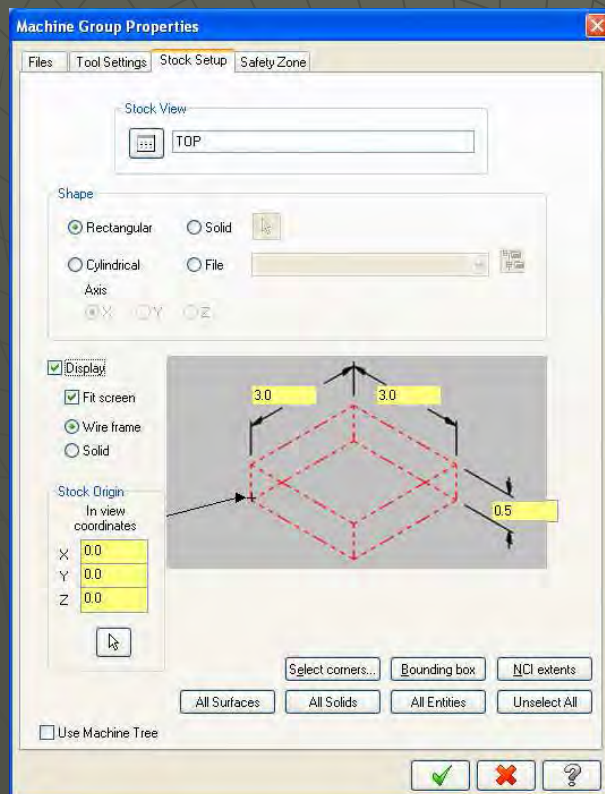
We will use *Mill_1_ 2012handout* in 7928 folder and follow follow the steps to set up stock size & stock origin

The project: Pivot Clamp

Consider:

- How we will hold the part
- Type of operations
- Facing, contours, pockets, spot drilling, drilling, & type of milling
- Feeds, Speeds. Tool type
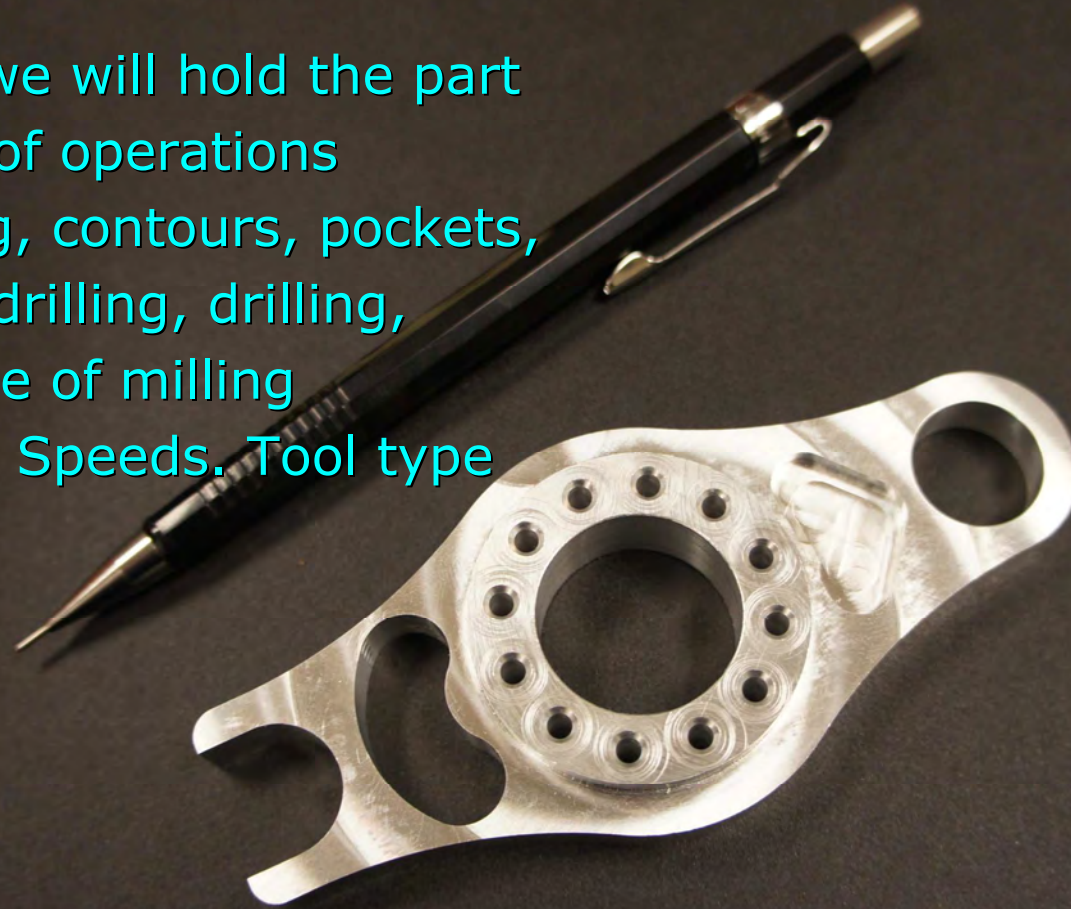
The Mill 1 handout provides the order & type of machining operations required, as well as tool selection & recommended feeds & speeds:
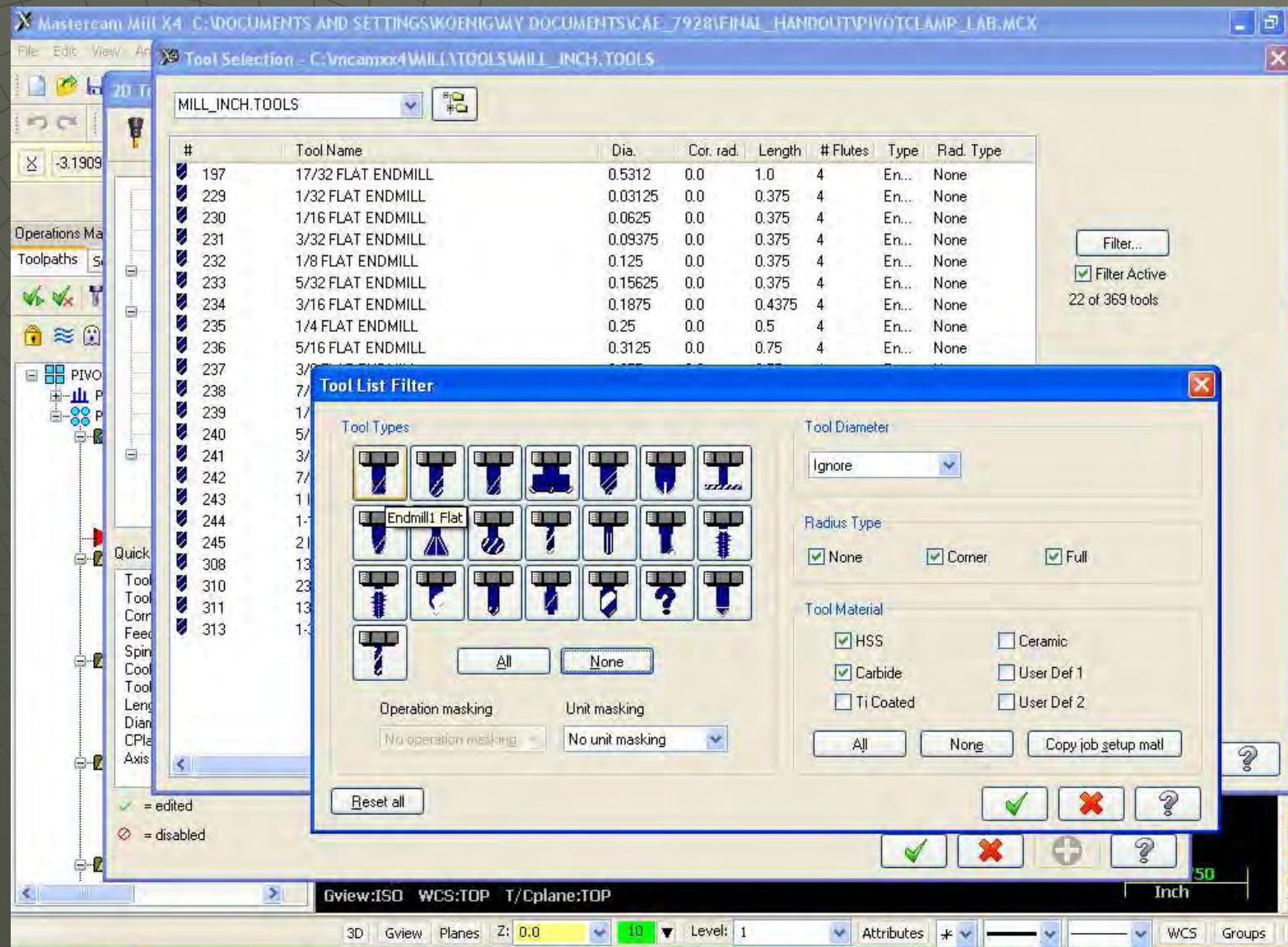(note the location of the origin in Mastercam)
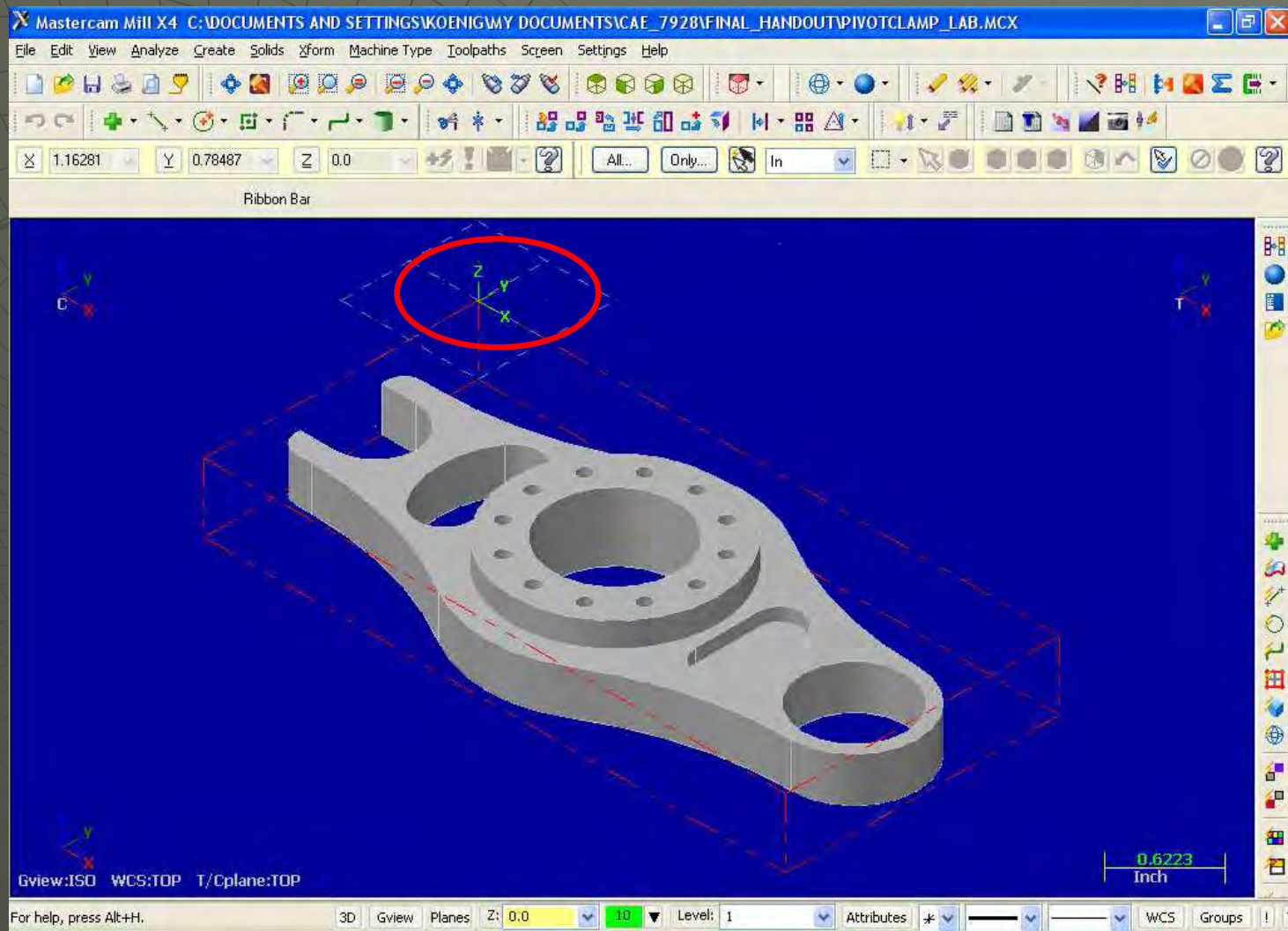
Machining steps:
Tip: *Think about rough cuts & finish cuts*

In Mastercam the
Operations Manager
shows the machining
operations, parameters,
geometry & toolpath- we will
create these operations.
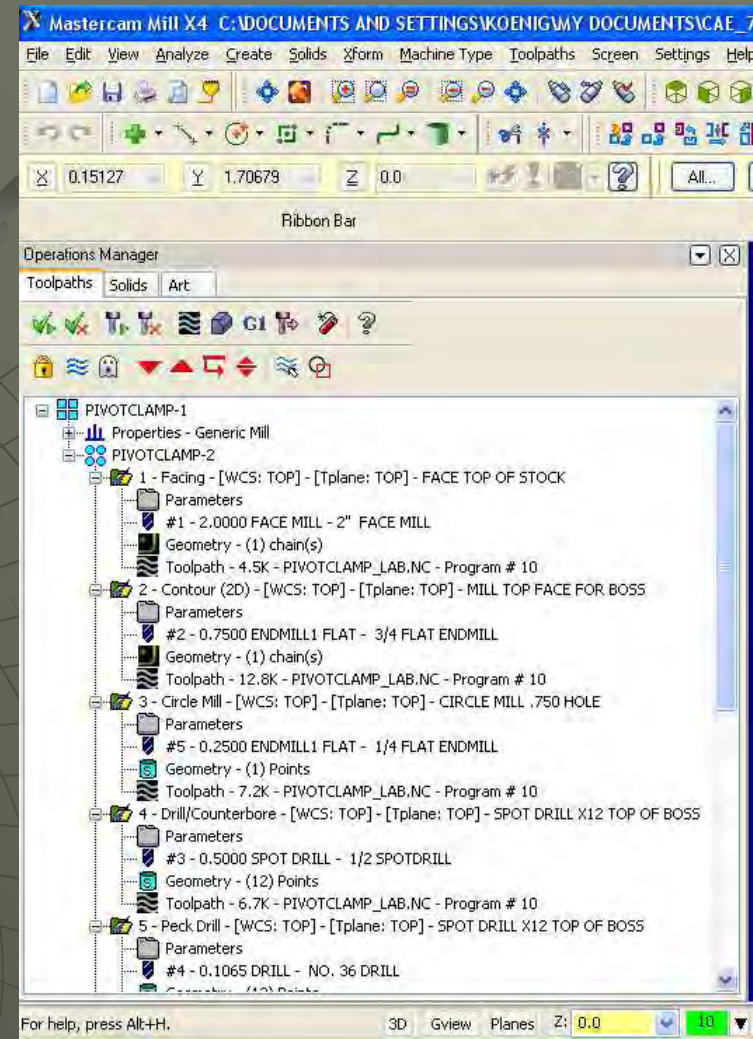
Select tools from the library:
Tip: Filter, None, then select the tool required.

Enter parameters as show in handout.

Tip: *Always enter a clear comment for each operation, this will appear in the code.*

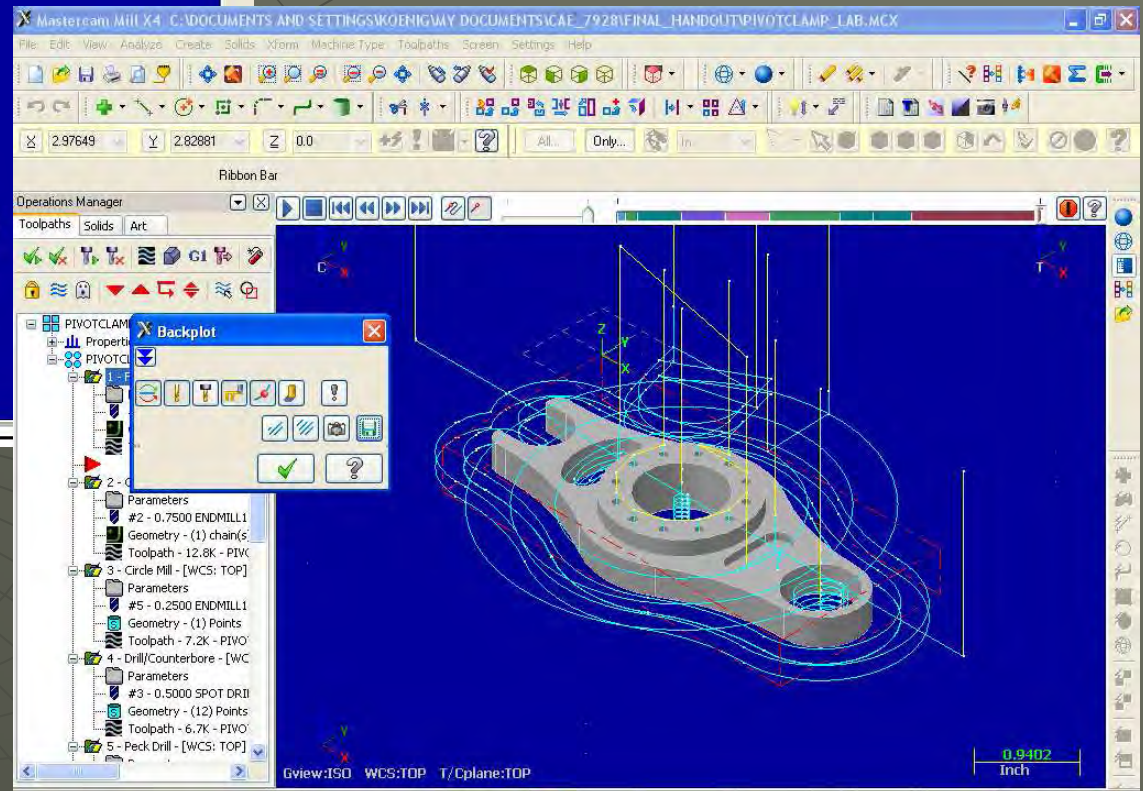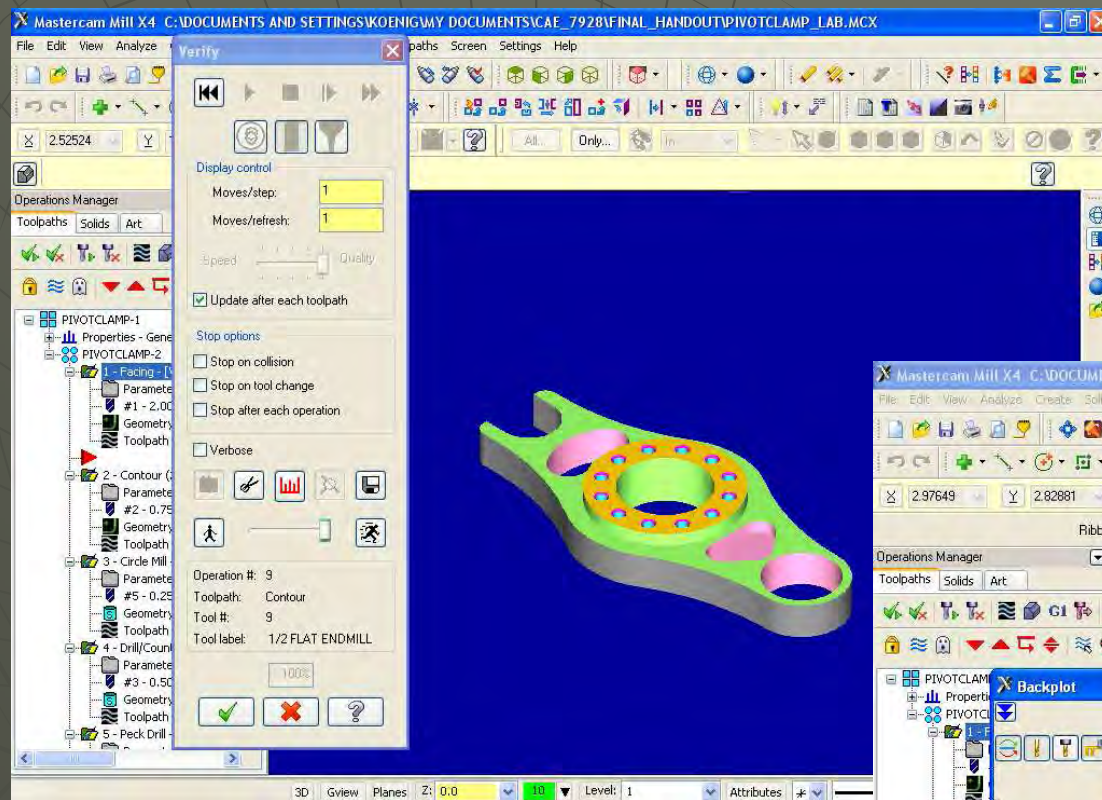# Enter the parameters for each machining operation in Mastercam
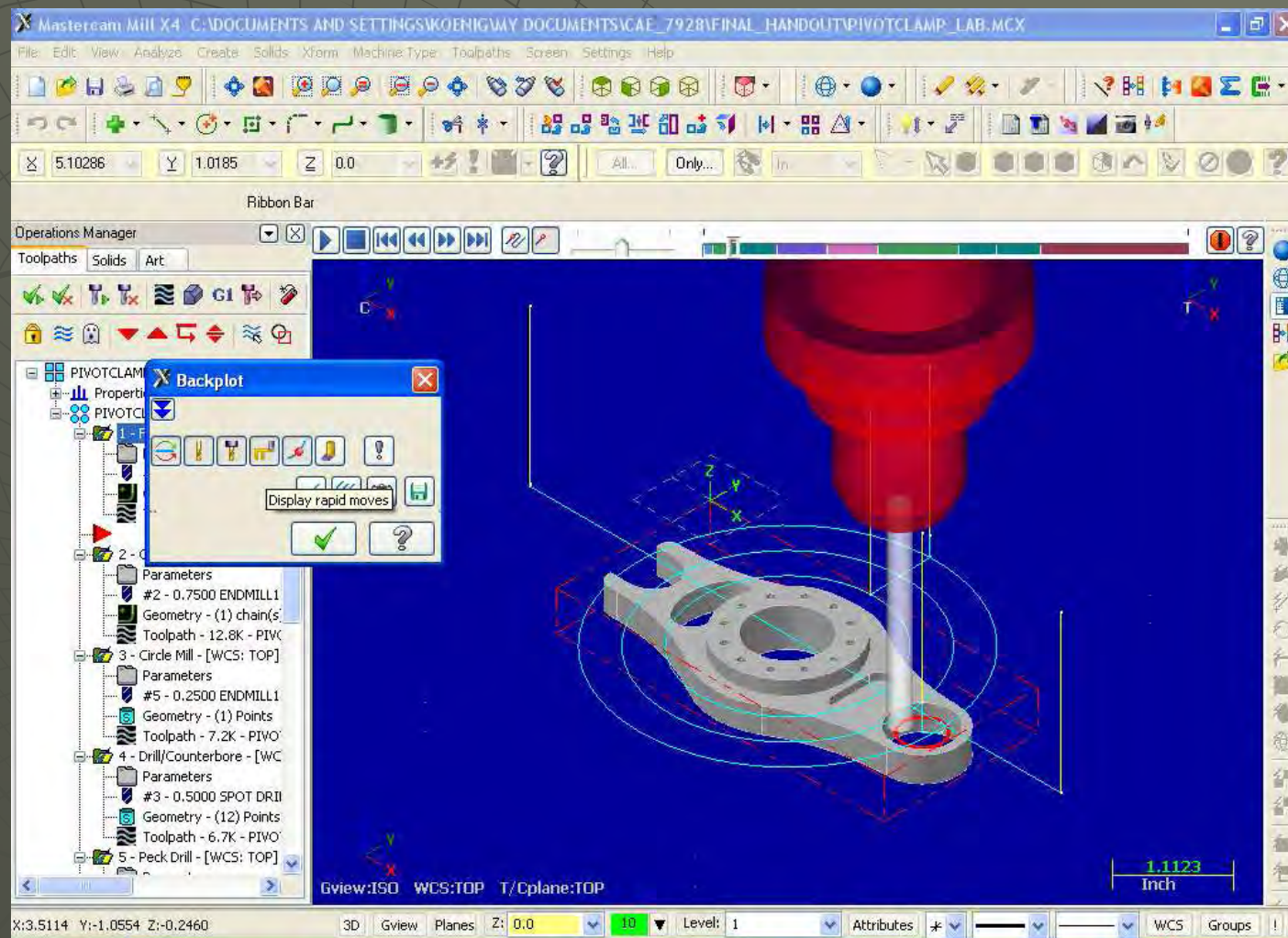*Tip: Be sure proper clearance is given along all axis around all work holding devices!*
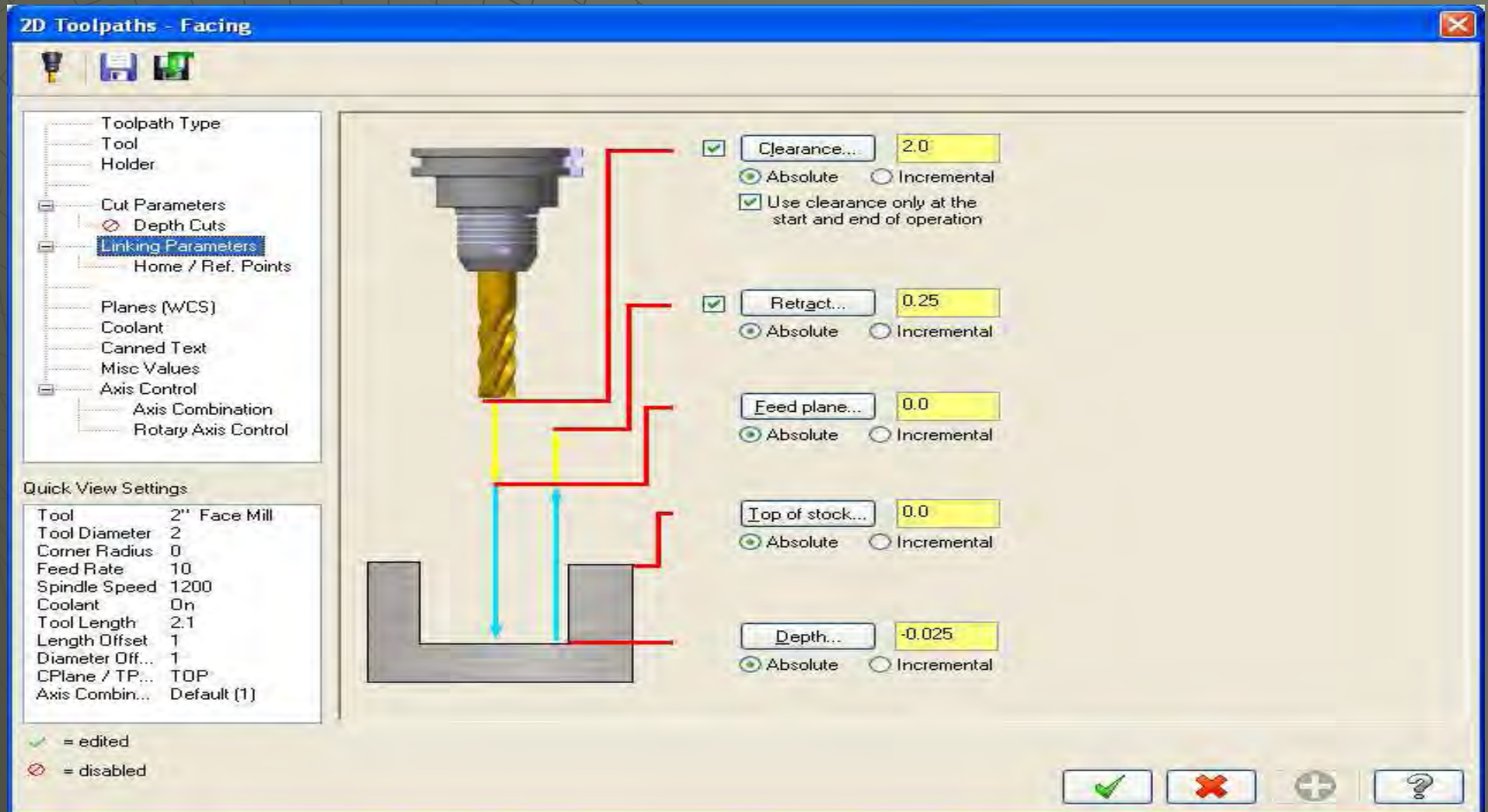
Use Verify & Backplot to check toolpaths, depth cuts, machining direction and more…

Tip: "Make your *virtual mistakes* in Mastercam"

Backplot will show the detailed machining steps created in each operation, tool, toolholder, rapids moves & more.
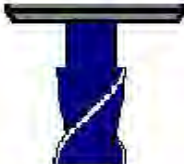
Generate a detailed Tool List

Generate a detailed Setup Sheet

Post the G Code to the post processor

Tip: Learn how to read code and check your work!

G Codes Preparatory Functions

HAAS C.N.C. MILL
PREPARATORY FUNCTIONS

MODAL
NON-MODAL
DEFAULT *
OPTIONAL **

Code	Description
G00*	RAPID POSITIONING MOTION (X,Y,Z,A,B) (SETTING 10, 56, 101)
G01	LINEAR INTERPOLATION MOTION (X,Y,Z,A,B,F)
G02	CIRCULAR INTERPOLATION MOTION CW (X,Y,Z,A,I,J,K,R,F)
G03	CIRCULAR INTERPOLATION MOTION CCW (X,Y,Z,A,I,J,K,R,F)
G04	DWELL (P) (P=seconds"."milliseconds)
G09	EXACT STOP, NON-MODAL
G10	PROGRAMMABLE OFFSET SETTING (X,Y,Z,A,L,P,R)
G12	CW CIRCULAR POCKET MILLING (Z,I,K,Q,D,L,F)
G13	CCW CIRCULAR POCKET MILLING (Z,I,K,Q,D,L,F)
G17*	CIRCULAR MOTION XY PLANE SELECTION (G02 or G03) (SETTING 56)
G18	CIRCULAR MOTION ZX PLANE SELECTION (G02 or G03)
G19	CIRCULAR MOTION YZ PLANE SELECTION (G02 or G03)
G20*	VERIFY INCH COORDINATE POSITIONING (SETTING 9, set to INCH)
G21	VERIFY METRIC COORDINATE POSITIONING (SETTING 9 set to METRIC)
G28	MACHINE ZERO RETURN THRU REF. POINT (X,Y,Z,A,B) (SETTING 108)
G29	MOVE TO LOCATION THROUGH G28 REF. POINT (X,Y,Z,A,B)
G31**	FEED UNTIL SKIP FUNCTION (X,Y,Z,A,B,F)
G35**	AUTOMATIC TOOL DIAMETER MEASUREMENT (D,H,Z,F)
G36**	AUTOMATIC WORK OFFSET MEASUREMENT (X,Y,Z,A,B,I,J,K,F)
G37**	AUTOMATIC TOOL LENGTH MEASUREMENT (D,H,Z,F)
G40*	CUTTER COMP CANCEL G41/G42/G141 (X,Y)
G41	2D CUTTER COMPENSATION, LEFT (X,Y,D) (SETTING 40, 43, 44, 58)
G42	2D CUTTER COMPENSATION, RIGHT (X,Y,D) (SETTING 40, 43, 44, 58)
G43	TOOL LENGTH COMPENSATION+ (H,Z) (SETTING 15)
G44	TOOL LENGTH COMPENSATION- (H,Z) (SETTING 15)
G47	TEXT ENGRAVING (X,Y,Z,R,I,J,P,E,F,) (Macro Variable 599 Change Serial #)
G49*	TOOL LENGTH COMPENSATION CANCEL G43/G44/G143 (SETTING 56)
G50*	SCALING G51 CANCEL (SETTING 56)
G51**	SCALING (X,Y,Z,P) (SETTING 71)
G52	WORK OFFSET POSITIONING COORDINATE (SETTING 33, YASNAC)
G52	GLOBAL WORK COORDINATE OFFSET SHIFT (SETTING 33, FANUC)
G52	GLOBAL WORK COORDINATE OFFSET SHIFT (SETTING 33, HAAS)
G53	MACHINE ZERO XYZ POSITIONING, NON-MODAL (X,Y,Z,A,B)
G54*	WORK OFFSET POSITIONING COORDINATE #1 (SETTING 56)
G55	WORK OFFSET POSITIONING COORDINATE #2
G56	WORK OFFSET POSITIONING COORDINATE #3
G57	WORK OFFSET POSITIONING COORDINATE #4
G58	WORK OFFSET POSITIONING COORDINATE #5
G59	WORK OFFSET POSITIONING COORDINATE #6
G60	UNI-DIRECTIONAL POSITIONING (X,Y,Z,A,B) (SETTING 35)
G61	EXACT STOP, MODAL (X,Y,Z,A,B)
G64*	EXACT STOP G61 CANCEL (SETTING 56)
G65**	MACRO SUB-ROUTINE CALL
G68**	ROTATION (G17,G18,G19,X,Y,Z,R) (OPTION) (SETTING 72, 73)
G69*	ROTATION G68 CANCEL (SETTING 56)
G70	BOLT HOLE CIRCLE with a CANNED CYCLE (I,J,L)
G71	BOLT HOLE ARC with a CANNED CYCLE (I,J,K,L)
G72	BOLT HOLES ALONG AN ANGLE with a CANNED CYCLE (I,J,L)
G73	HIGH SPEED PECK DRILL CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,R,L,F) (SETTING 22, 52)
G74	REVERSE TAPPING CANNED CYCLE (X,Y,A,B,Z,J,R,L,F) (SETTING 130, 133)
G76	FINE BORING CANNED CYCLE (X,Y,A,B,Z,I,J,P,Q,R,L,F) (SETTING 27)
G77	BACK BORE CANNED CYCLE (X,Y,A,B,Z,I,J,Q,R,L,F) (SETTING 27)
G80*	CANCEL CANNED CYCLE (SETTING 56)
G81	DRILL CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G82	SPOT DRILL / COUNTERBORE CANNED CYCLE (X,Y,A,B,Z,P,R,L,F)
G83	PECK DRILL DEEP HOLE CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,R,L,F) (SETTING 22, 52)
G84	TAPPING CANNED CYCLE (X,Y,A,B,Z,J,R,L,F) (SETTING 130, 133)
G85	BORE IN~BORE OUT CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G86	BORE IN~STOP~RAPID OUT CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G87	BORE IN~MANUAL RETRACT CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G88	BORE IN~DWELL~MANUAL RETRACT CANNED CYCLE (X,Y,A,B,Z,P,R,L,F)
G89	BORE IN~DWELL~BORE OUT (X,Y,A,B,Z,P,R,L,F)
G90*	ABSOLUTE POSITIONING COMMAND
G91	INCREMENTAL POSITIONING COMMAND (SETTING 29)
G92	GLOBAL WORK COORDINATE SYSTEM SHIFT (FANUC) (HAAS) (SETTING 33)
G92	SET WORK COORDINATE VALUE (YASNAC) (SETTING 33)
G93	INVERSE TIME FEED MODE ON
G94*	INVERSE TIME FEED MODE OFF / FEED PER MINUTE ON (SETTING 56)
G95	FEED PER REVOLUTION (SETTING 9, 56)
G98*	CANNED CYCLE INITIAL POINT RETURN (SETTING 56)
G99	CANNED CYCLE "R" PLANE RETURN
G100	MIRROR IMAGE G101 CANCEL
G101	MIRROR IMAGE (X,Y,Z,A,B) (SETTING 45, 46, 47, 48, 80)
G102	PROGRAMMABLE OUTPUT TO RS-232 (X,Y,Z,A,B)
G103	LIMIT BLOCK LOOKAHEAD (P0-P15 for number control looks ahead)
G107	CYLINDRICAL MAPPING (X,Y,Z,A,Q,R,)
G110-G129	WORK OFFSET POSITIONING COORDINATE #7-26
G136**	AUTOMATIC WORK OFFSET CENTER MEASUREMENT
G141	3D+ CUTTER COMPENSATION (X,Y,Z,I,J,K,D,F)
G143**	5-AXIS TOOL LENGTH COMPENSATION+ (X,Y,Z,A,B,H) (SETTING 15, 117)
G150	GENERAL PURPOSE POCKET MILLING (X,Y,P,Z,I,J,K,Q,D,R,L,S,F)
G153**	5-AXIS HIGH SPEED PECK DRILL CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,E,L,F) (SETTING 22)
G154**	SELECT WORK OFFSET POSITIONING COORDINATE P1-99 (P)
G155**	5-AXIS REVERSE TAPPING CANNED CYCLE (X,Y,A,B,Z,J,E,L,F)
G161**	5-AXIS DRILL CANNED CYCLE (X,Y,A,B,Z,E,L,F)
G162**	5-AXIS SPOT DRILL / COUNTERBORE CANNED CYCLE (X,Y,A,B,Z,P,E,L,F)
G163**	5-AXIS PECK DRILL CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,E,L,F) (SETTING 22)
G164**	5-AXIS TAPPING CANNED CYCLE (X,Y,A,B,Z,J,E,L,F)
G165**	5-AXIS BORE IN, BORE OUT CANNED CYCLE (X,Y,A,B,Z,E,L,F)
G166**	5-AXIS BORE IN, STOP, RAPID OUT CANNED CYCLE (X,Y,A,B,Z,E,L,F)
G169**	5-AXIS BORE IN, DWELL, BORE OUT (X,Y,A,B,Z,P,E,L,F)
G174	NON-VERTICAL RIGID TAPPING CCW (X,Y,Z,F)
G184	NON-VERTICAL RIGID TAPPING CW (X,Y,Z,F)
G187	ACCURACY CONTROL FOR HIGH SPEED MACHINING (E) (SETTING 85)
G188	G188 GET PROGRAM FROM PST (Program Schedule Table)

M Codes – Misc. Functions

HAAS C.N.C. MILL
MISCELLANEOUS FUNCTIONS

OPTION**

Code	Function	Code	Function
M00	PROGRAM STOP (SETTING 39, 42)	M50**	EXECUTE PALLET CHANGE (P) (SETTING 121 thru 129)
M01	OPTIONAL PROGRAM STOP (SETTING 17, 39)	M51-M58	OPTIONAL USER M CODE SET
M02	PROGRAM END (SETTING 39)	M59	OUTPUT RELAY SET (N)
M03	SPINDLE ON CLOCKWISE (S) (SETTING 144)	M61-M68	OPTIONAL USER M CODE CLEAR
M04	SPINDLE ON COUNTERCLOCKWISE (S) (SETTING 144)	M69	OUTPUT RELAY CLEAR (N)
M05	SPINDLE STOP	M75	SET G35 OR G136 REFERENCE POINT
M06	TOOL CHANGE (T) (SETTING 42, 87, 155)	M76	CONTROL DISPLAY INACTIVE
M08	COOLANT ON (SETTING 32)	M77	CONTROL DISPLAY ACTIVE
M09	COOLANT OFF	M78	ALARM IF SKIP SIGNAL FOUND
M10**	4th AXIS BRAKE ON	M79	ALARM IF SKIP SIGNAL NOT FOUND
M11**	4th AXIS BRAKE RELEASE	M80**	AUTOMATIC DOOR OPEN (SETTING 131)
M12**	5th AXIS BRAKE ON	M81**	AUTOMATIC DOOR CLOSE (SETTING 131)
M13**	5th AXIS BRAKE RELEASE	M82	TOOL UNCLAMP
M16	TOOL CHANGE (T) (same as M06)	M83**	AUTO AIR JET ON
M17**	APC PALLET UNCLAMP and OPEN APC DOOR	M84**	AUTO AIR JET OFF
M18**	APC PALLET CLAMP and CLOSE DOOR	M86	TOOL CLAMP
M19	ORIENT SPINDLE (P,R values optional)	M88**	COOLANT THROUGH SPINDLE ON (SETTING 32)
M21-M28	OPTIONAL USER M CODE INTERFACE WITH M-FIN SIGNAL	M89**	COOLANT THROUGH SPINDLE OFF (SETTING 32)
M30	PROGRAM END AND RESET (SETTING 2, 39, 56, 83)	M95	SLEEP MODE
M31	CHIP AUGER FORWARD (SETTING 114, 115)	M96	JUMP IF NO INPUT (P,Q)
M33	CHIP AUGER STOP	M97	LOCAL SUB-PROGRAM CALL (P,L)
M34	COOLANT SPIGOT POSITION DOWN, INCREMENT (+1)	M98	SUB-PROGRAM CALL (P,L)
M35	COOLANT SPIGOT POSITION UP, DECREMENT (-1)	M99	SUB/LOCALSUB-PROGRAM / RETURN OR LOOP (P) (SETTING 118)
M36**	PALLET PART READY (P)	M101	MOM (Minimum Oil Machining) CANNED CYCLE MODE (I)
M39	ROTATE TOOL TURRET (T#) (SETTING 86)	M102	MOM MODE (I,J)
M41	SPINDLE LOW GEAR OVERRIDE	M103	MOM MODE CANEL
M42	SPINDLE HIGH GEAR OVERRIDE	M109**	INTERACTIVE USER INPUT (P)

All **M** codes are effective or cause an action at the end of the block and only one **M** code is allowed in each block.

Haas Automation Inc.

Updated on 1-01-06

Project Report Requirements

- Cover page
- SolidWorks detail drawing: Pivot Clamp
- Print all toolpaths page Mastercam
- Print tool list page Mastercam
- Print set up sheet page Mastercam
- Print the 1st page of the NC Code & Annotate the major G & M code operations on this page – see G & M Code sheets provided)

Some parts modeled in SolidWorks (CAD) toolpaths created using Mastercam (CAM)...&?
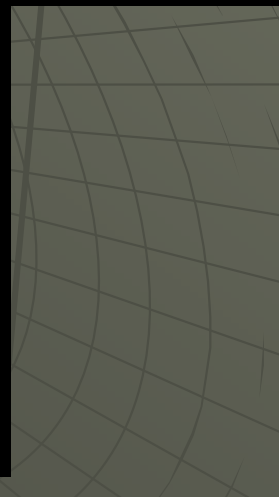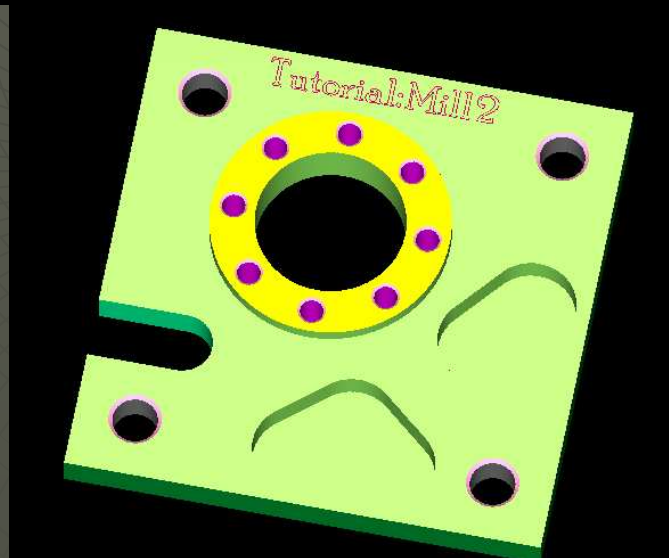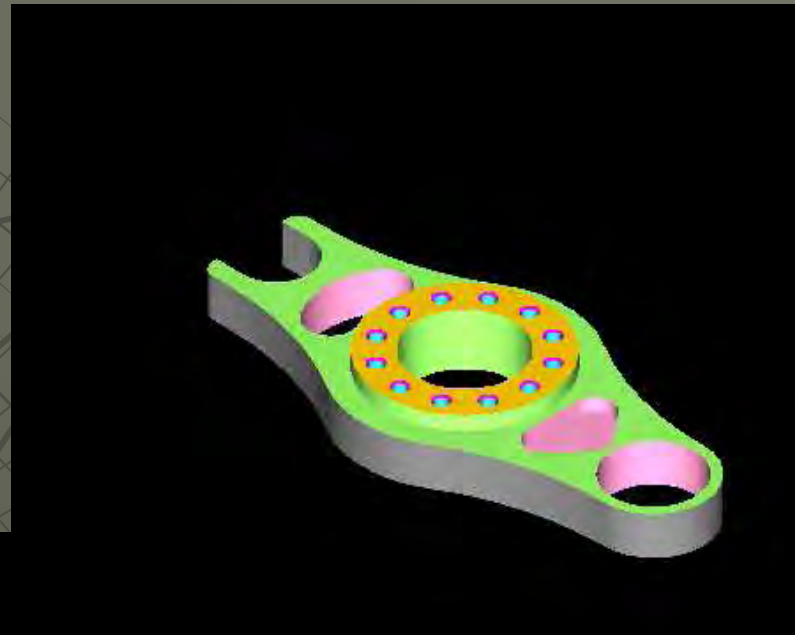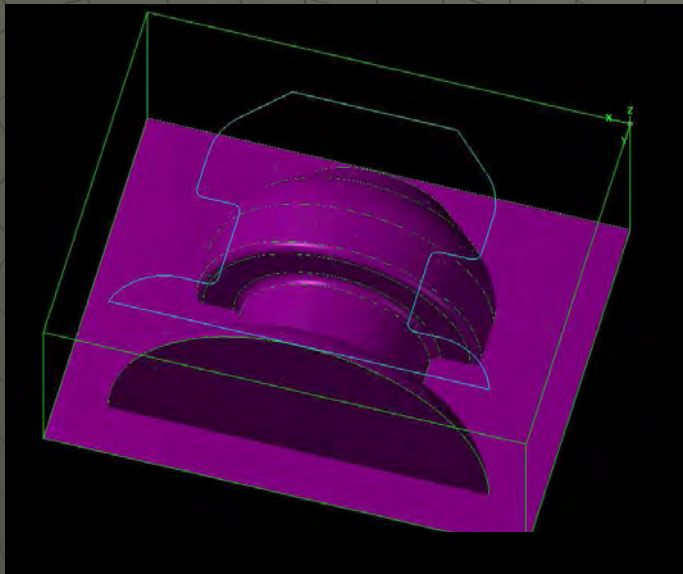
Oh…one last thing!

When opening Mastercam you will need to:

Select:

- Start
- All Programs
- Mastercam X4
- Nhasp X
- Select NetHASP button
- Read
- Click Activate Licenses button
- OK